10th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2006, Barcelona - Lloret de Mar, Spain, 11-15 September, 2006

FEM SIMULATION OF TIE-ROD TENSILE TEST

MSc. Ismar Alagić
REZ-RDA Central BiH / University of Zenica,
Zenica
Bosnia and Herzegovina
ismar@rez.ba

ABSTRACT

Advanced solid modelling in Algor FEM software is ideally suited for testing functionality of automobile parts.
This article shows that Finite Element Method (FEM) helps designer asses the effects of flexible components on full system performance, improve the accuracy of simulations and thus bring it closer to the system-level design. The conducted research has begun with creation of 3D-CAD solid approximate model in the form of a multi-body system, after that solid mesh was generated where all meshed elements assumed to be perfectly rigid, and in final stage of testing finite element analysis was performed using Algor software package.
Tie rod, a part of steering system, is primarily used in automobile industry in wide variety of exploitation conditions during actual driving conditions and road test.
In order to conduct the tensile test of tie bar from housing of tie rod assembly (tensile force, F=30000 N) designed for assembling wheel transmission system of passenger vehicles, finite element calculation has been carried out using Algor software. Sufficiently accurate stress distribution and displacement distribution of tie rod assembly have been obtained through the whole tensile test.
The accuracy of the simulation results after unloading is compared to experimental results. The experiment is performed by internal control device MR 96.
Key words: Finite element method (FEM), Tie rod, Tensile test, Algor.

1. INTRODUCTION

In automobiles a tie rod is part of the steering mechanism. The tie rods connect the centre link to the steering knuckle on cars with conventional suspension systems and recirculation ball steering gears. The tie rod transmits force from the steering centre link or the rack gear to the steering knuckle, causing the wheels to turn. The real problem with the tie rod assembly is its strength and durability. The tie rod assembly consists of four following separate parts: tie bar, housing, bushing and bellow. Several methods have been proposed for estimating the actual tensile forces in tie rods.
It is imperative that tie rod operate reliably during exploitation and under severe heavy working conditions, since cars are entrusted with the care of precious human life which would be endangered if an accident occurs. Tie rod as part of steering system is primarily used in automobile industry in wide variety of exploitation conditions during actual driving conditions [1].
This article shows that Finite Element Method (FEM) helps designer asses the effects of flexible components on full system performance, improve the accuracy of simulations and thus bring it closer to the system-level design. The conducted research has begun with creation of 3D-CAD solid approximate model in the form of a multi-body system, after solid mesh was generated where all meshed elements assumed to be perfectly rigid, and in final stage of testing finite element analysis was performed using Algor software package.

2. DESCRIPTION OF TIE ROD FE-MODEL

In order to conduct the tensile test of tie bar from housing of tie rod assembly (tensile force, F=30000 N) designed for assembling wheel transmission system of passenger vehicles, finite element calculation has been carried out using Algor software. The aim of this research is simulation of tensile test (tensile force, magnitude 30 000 N) using finite element method. The accuracy of finite element model depends on the assumptions made and the correlation between the computer models and testing application. The constructing model depends on material properties as well as testing conditions and testing equipment.

Integrated system of three components, firstly CAD three dimensional design work is shown in figure 1 (performed by Autodesk Mechanical Desktop), Finite element (FE) model (software Algor Version 16), and on Mechanical Event Simulation (MES) as a level of motion simulation, improve the reliability of load prediction by using FE-model [2] . This model allows obtaining optimal structure characteristics of the tie rod in the short time and with high accuracy.

Design documentation requires the following allowed technical characteristics:

a) Maximum allowed displacement 0,5 mm;
b) Maximum allowed warp angle 38+6°;
c) Ultimate stress (material of tie bar) 1100-1300 MPa.

a) Tie rod-tie rod end assembly; *b) Tie rod assembly;* *c) Tie rod.*

Figure 1. Three-dimensional model.

Figure 2. Mesh of tie rod assembly.

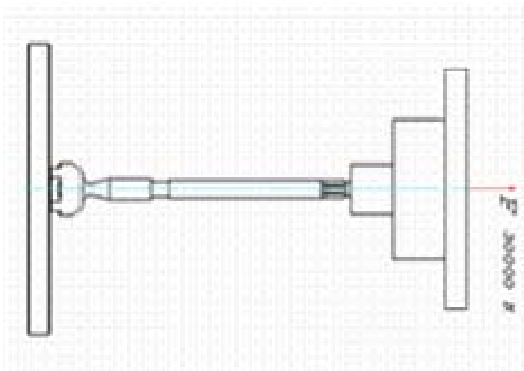

Figure 3. Experiment performed by device MR 96.

Numerical calculation is performed with aid of the FEM (software Algor). Three dimensional models of tie rod consisted of tetrahedral solid elements. The models were built with Algor R Version 16, Static Stress with Linear Material Models. Total number of finite elements was 85 875 elements for all analyzed models. Tensile loads of tie rod were used as FEM boundary conditions. Tensile load was used for tie rod stress, strain and displacement validation. All necessary input information regarding FEM analysis is shown below.

Table 1. Material properties.

Material	Mass density, kg/m^3	Modulus of elasticity, MPa	Poisson's ratio	Shear modulus of elasticity, MPa
Steel, ASTM-A242	7854,8	199950	0,29	77221
Plastics Nylon, Type 6/6	1143,5	2757,9	0,35	1021,4

Table 2. Technical characteristic of model elements.

Part ID	Part Name	Material Name	Element Type	Number of elements
1	Tie bar	Steel (ASTM-A242)	tetrahedral	43002
2	Bushing	Plastics Nylon, Type 6/6	tetrahedral	10043
3	Housing	Steel (ASTM-A242)	tetrahedral	20407
4	Bellow	Plastics Nylon, Type 6/6	tetrahedral	12423

Table 3. Surface Force.

ID	Description	Part ID	Surface ID	Magnitude	V_x	V_y	V_z
1	Tensile force	3	29	30000	1	0	0

3. FE-SIMULATION RESEARCH AND EXPERIMENTAL RESULTS

Control device MR 96 is the horizontal testing machine suitable for tensile test of long materials such as steering tie rods. The tie rod is loaded in tension. The tensile test on three tie rod sample, of 21 mm inner diameter and 26 mm outer diameter and 329 mm of length, have provided the following results presented in table 4. During the tensile test, all results were recorded. The tensile force is calculated using: $F_t = A \cdot E \cdot \varepsilon_t$, where: A is the cross section, E is the Young modulus of the tie rod and ε_t is the tension strain.

To prevent any damage, each of three tie rods (three samples) will be tested at 120% of its maximum exploitation loads (tensile load, magnitude 24 kN). The tie rods have been pulled at 30 000 N. No damages have been noticed.

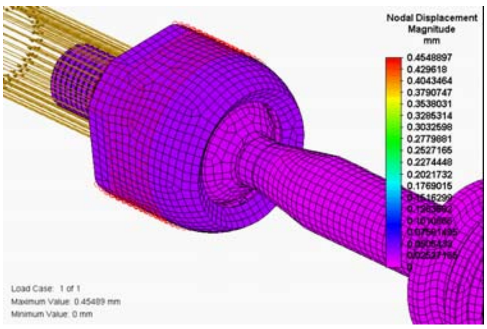

Figure 4. FEM model –displacement (magnitude).

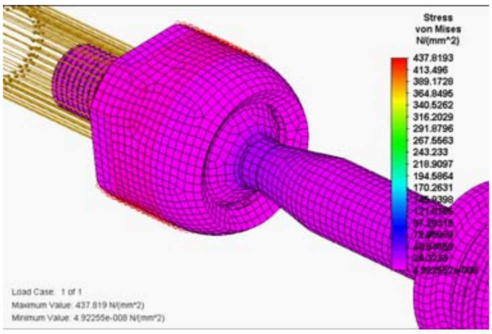

Figure 5. FEM model-stress distribution.

The distribution, direction and value of the tensile force presumed in the analysis were similar to value and the direction, which were recorded at the maximal exploitation load. As a matter of fact, the total produced tensile force F=30000 N, was approximately 20% above the value of maximum exploitation load applied during actual driving conditions. Therefore testing procedures provides reliably assumption of this problem.

A 3D-CAD study has been performed by FEM with Algor R Version 16. The results of FEM simulation are provided in this document (figure 4, 5, 6 and 7). The safety margin of the tie rod is defined as material strength/maximum stress. The experimental tensile test of tie bar from housing of tie rod assembly is performed by control device MR 96 (see table 4). The deformation achieved as result of finite element analysis is similar to the results of the tensile test performed by control device MR 96. The maximum appeared displacement (0,455<allowed value 0, 5 mm, see figure 4) and

maximum stress value is lower than ultimate stress allowed by documentation request (437,8<1100 MPa, see figure 5). Also, analysis of strain distribution (see figure 6) and warp angle (41,4<allowed value 44°, see figure 7) through the whole tensile test showed to us lower angle than allowed value required by design documentation.

Table 4. Experimental results performed by control device MR 96, tensile force F=30 000 N.

Sample	I (mm)	II (mm)	III (mm)
Inner diameter of tie bar before tensile test	ϕ21,40	ϕ21,40	ϕ21,45
Inner diameter of tie bar after tensile test/elongation	ϕ21,42 / 0,02	ϕ21,45 /0,05	ϕ21,50 /0,05
Outer diameter of tie bar before tensile test	ϕ26,30	ϕ26,30	ϕ26,30
Outer diameter of tie bar after tensile test/ elongation	ϕ26,40 / 0,1	ϕ26,32 /0,02	ϕ26,35 /0,05

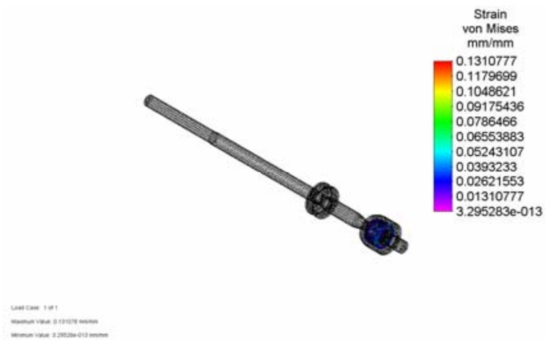

Figure 6. FEM model - strain distribution.

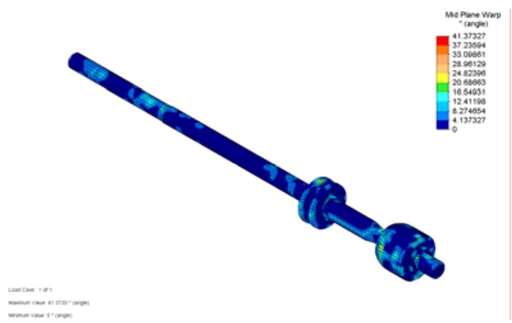

Figure 7. FEM model - warp angle.

4. CONCLUSION

On the basis of research results, it is possible to conclude the following:

- The conducted research has begun with creation of 3D-CAD solid approximate model in the form of a multi-body system, after that solid mesh was generated where all meshed elements assumed to be perfectly rigid, and in final stage of testing finite element analysis was performed using Algor software package.
- From the presented results we can conclude that the distribution of deformation and stress do not exceed the upper limit value and that there are neither damages nor surface defects after performed tensile test.
- The results of tensile test performed by control device MR 96 were closer to the results of FEM simulation.
- Using FEM made possible to predict the whole tensile test of tie rod assembly.
- The correctness and accuracy of computed results is still dependent on the selection related to various modelling parameters. Some of the most important aspects, such as boundary conditions or correct mesh and type of elements are performing a decisive role in achieving of correct results.
- The mentioned conclusion is only valid for above defined working conditions and incorrigible estimated value of tensile force (F=30000 N).

5. REFERENCE

[1] Alagić I.: Design and testing of ball joint No. 181160010 using FEA, 8th International Research/Expert Conference TMT 2004, Neum, B&H, 15-19. September, 2004.

[2] Alagic I. :"Using the Finite Element Method (FEM) in order to Optimize of Clamping Force for Jaw-Chucks during Machining Process of the Brake Disc", 9th International Research/Expert Conference TMT 2005, Antalya, Turkey, 26-30. September, 2005.